OPTIMIZATION OF COMPOSITE STRUCTURE BASED ON FRACTURE ANALYSIS

Abdelnaser Abdusalam Elayeb[1*], Aleksandar Grbovic[2], Martina Balac[3]

[1]University of New Brunswick, Faculty of Engineering, The Department of Mechanical Engineering, Canada

[2]University of Belgrade, Faculty of Mechanical Engineering, Department for Aeronautics, 11000 Belgrade, Serbia

[3]University of Belgrade, Faculty of Mechanical Engineering, Department for Process Engineering and Environment Protection, 11000 Belgrade, Serbia

*Corresponding author e-mail: agrbovic@mas.bg.ac.rs

Abstract

Over the last three decades, the use of composites in aviation structures has been increasing. The previous aircraft structure was mostly made of aluminum alloy, and the composites were limited to a level two structure. With increasing experience and confidence, the use of these materials has been extended to major structures. Moreover, composites used in vehicles industry can reduce weight and operating costs, so it is believed that their use will increase further in the near future. The research scope of this study was to develop the optimal design of a composite structure in the terms of maximum vehicle's life. To achieve this, several composite designs of a vehicle body were analyzed in Ansys Fluent to get the loads and stresses that can be expected during the service. Then, Helius PFA software for progressive failure analysis has been used. It provides powerful tools for enhanced finite element analysis of composite structures. The validity of the developed model has been investigated and compared with an experimental data collected in the wind tunnel for some composite specimens and for a developed 3D model. Moreover, a comparison study, between experimental and simulation work, was also performed with selected design cases to demonstrate the potential advantages of using the optimal composite structure to maintain fatigue long-life structures.

Keywords

Composite structures; fatigue life; finite element analysis; fracture analysis